REMARKS

This is responsive to the Office Action mailed July 19, 2004, relating to the above-identified application. Applicant respectfully traverses and requests reexamination and reconsideration. Claims 1-4 and 6-20 remain pending.

Claim Amendments

Rejection of Claims Under 35 U.S.C. 103(a)

Claims 1 and 4

Claims 1 and 4 stand rejected under 35 U.S.C. § 103(a) as being unpatentable over U.S. Patent No. 6,329,954 (hereinafter referred to as "Fuchs"). Applicant respectively traverses and submits the present rejection as being improper and should be withdrawn.

For the sake of brevity, Applicants respectfully resubmit the previously offered position regarding the teachings of Fuchs, as noted in the response filed on December 23, 2003:

> As recognized by one having ordinary skill in the art, a printed circuit board does not inherently contain a ground plane, but rather may contain traces on both sides of the pcb. As noted above, the full teachings of Fuchs regarding the pcb consists of: "the board 132 also carries devices electrically connected to the antenna elements and providing appropriate amplification of the signals received by the elements." (see col. 4, lines 15-18).

Moreover, in the interest of gravity, Applicant respectfully submits the previous offered position regarding the teachings of Fuchs, as noted in the Response filed on June 7, 2004:

> As noted above, Fuchs is silent regarding the structure of the pcb 132, but rather relies on the general statement that the "board is otherwise generally well-known to those skilled in the

> art." (See column 4, lines 18-19). Therefore, at best, Fuchs discloses the pcb may be a standard pcb board, but Fuchs fails to teach or suggest the monopole array being electrically coupled together ... The abstract teaches the alignment of the monopoles, but fails to teach or suggest the connectivity of the monopole elements.

Claim 1, as previously amended, requires, among other things, a ground plane on an upper surface of a printed circuit board. The Examiner asserts that a pcb by definition is a dielectric with a ground plane on one side and circuitry on the opposing side. Applicant notes that the Examiner appears to have taken an official notice without documentary evidence to support such a conclusion. Applicant submits that there is no evidence in the record to support the Examiner's conclusion and requests a specific showing with documentary evidence to support the Examiner's position.

As contrary evidence thereof, Applicant submits that by definition a printed circuit board does not have a ground plane. Rather, an accepted definition of a printed circuit board of those skilled in the art, is a flat plate or base of insulating material containing a pattern of conducting material which becomes an electric circuit when components are attached and soldered to it. See Buban and Schmitt, *Technical Electricity and Electronics*, pp. 345-356 (1972) (attached as Exhibit A); Shrader, *Electronic Communication*, pp. 18-19 (1975) (attached as Exhibit B); http://www.pcwebopaedia.com/TERM/P/printed_circuit_board.html (attached as Exhibit C); and http://www.goldengategraphics.com/pcgloss.htm (attached as Exhibit D).

Moreover, Fuchs expressly discloses that the ground plane for its antenna system is the roof of an automobile (column 3, lines 47-48; column 4, lines 47-49; column 6, lines 35-36; and column 6, lines 49-51). Still further, Fuchs disclosed that the board 132 also carries devices electrically connected to the antenna elements and providing appropriate amplication of the signals received by the elements. However, Fuchs does not disclose whether such devices are on the upper surface, the lower surface or both surfaces of the printed circuit board. In view of the express teaching of Fuchs that the roof of the automobile serves as a ground plane, Applicant submits that the devices may be disposed on either or both sides of the board 132 in Fuchs as neither is a ground plane in view of the above teaching. To the extent the Examiner continues to rely on the asserted implications of Fuchs or knowledge asserted to be common in the art, Applicant requests a specific showing based on documentary evidence as to the basis of the Examiner's position. Consequently, Applicant submits that all of the limitations of Claim 1 are not met or suggested by Fuchs.

The Examiner asserts that Fuchs discloses each the monopoles being connected in column 5, lines 27-29. However, such statement by Fuchs fails to disclose the structure and function described with regard to the monopole array of the present invention.

Therefore, Applicant submits that Fuchs fails to disclose, teach or suggest all the claimed limitations of claims 1 and 4 when combined with the Examiner's citation of common knowledge and implication of positioning a ground plane on the upper surface of the pcb. Fuchs, in fact, fails to

disclose, teach or suggest the claimed ground plane, and provides an expressly different teaching regarding the ground plane in his antenna systems. Fuchs also fails to teach or suggest the connectivity of the monopole elements as claimed. Accordingly, the present rejection is improper. Reconsideration and withdrawal of the present rejection is respectfully requested.

Rejection of claims 2-3 and 6-20

Claims 2-3 and 6-20 stand rejected under 35 U.S.C. § 103(a) as being unpatentable over Fuchs in view of U.S. Patent No. 6,300,917 (hereinafter referred to as "Leisten"). Applicant respectively traverses and submits the present rejection as being improper and should be withdrawn.

Regarding claims 12, 16 and 20, Applicants respectfully resubmit the above-offered position regarding claim 1. As stated above, Fuchs fails to teach or suggest the ground plane as claimed and the structural and functional relationship of the monopole elements being electrically coupled via lower surface micro-strip traces. Therefore, the combination of Fuchs with Leisten will still produce a system failing to teach or suggest the ground plane disposed on the upper surface of the pcb and the structural and functional relationship of electrical coupling of monopole elements of the monopole array. Therefore, for at least the reasons stated above, it is submitted the present rejection is improper. Reconsideration and withdrawal is respectfully requested.

Regarding claims 2-3, 6-11, 13-15 and 17-19, Applicants respectfully resubmit the above offered position regarding claims 1, 12 and 16

respectively. It is submitted that claims 2-3, 6-11, 13-15 and 17-19 contain further patentable subject matter and are allowable not merely as being dependent upon allowable base claims. Therefore, reconsideration and withdrawal of the present rejection is respectfully requested.

CONCLUSION

Accordingly, Applicant respectfully submits that the claims are in condition for allowance and request that a timely Notice of Allowance be issued in this case. The Commissioner is hereby authorized to charge any underpayment or credit any overpayment to Deposit Account No. 22-0259 or any payment in connection with this communication, including any fees for extension of time, which may be required. The Examiner is invited to contact the below-listed attorney if the Examiner believes that a telephone conference will advance the prosecution of this application.

Respectfully submitted,

Dated: September 17, 2004

By: Michael J. Turgeon

Michael J. Turgeon
Reg. No. 39,404

Vedder, Price, Kaufman & Kammholz, P.C.
222 North LaSalle Street, Ste. 2400
Chicago, Illinois 60601-1003
(312) 609-7716

16. How can common pliers be used as a heat sink in soldering semiconductor components?

17. Describe how a copper-wire tip can be added to the tip of an ordinary soldering iron.

18. What is the recommended procedure when starting to wire a circuit from a schematic diagram?

19. For what purpose is insulating tubing, or "spaghetti," used?

20. For what purpose are rubber grommets used?

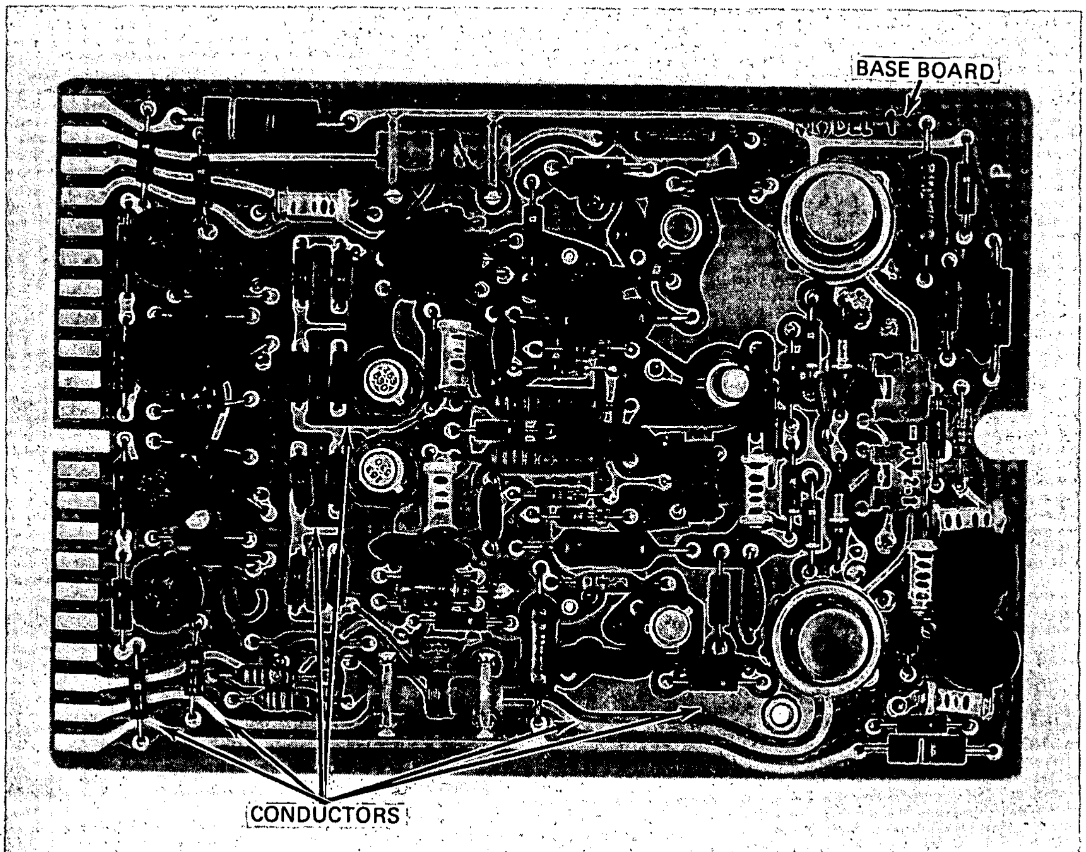
21. What is a power cord strain relief device and why is such a device used?

22. Describe a wire harness.

UNIT 31. Printed Circuits

The typical printed circuit assembly consists of components mounted upon a baseboard and soldered to thin strips of copper that are bonded to the base (Fig. 31-1). Although there are several different kinds of base materials used, two very common ones are phenolic and epoxy plastic. In addition to supporting the circuit assembly, the base material must provide adequate insulation and must be able to withstand high soldering and operating temperatures without

Fig. 31-1. Component side of a printed circuit assembly. (*Tektronix, Inc.*)



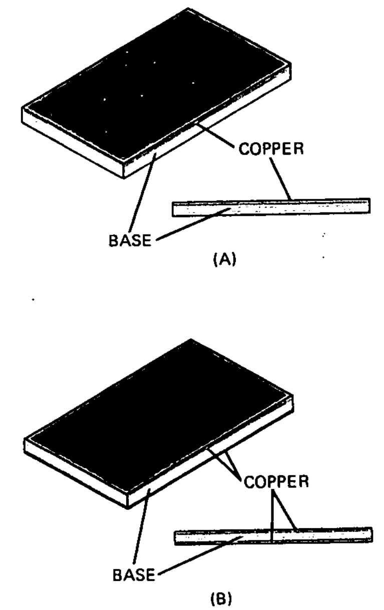


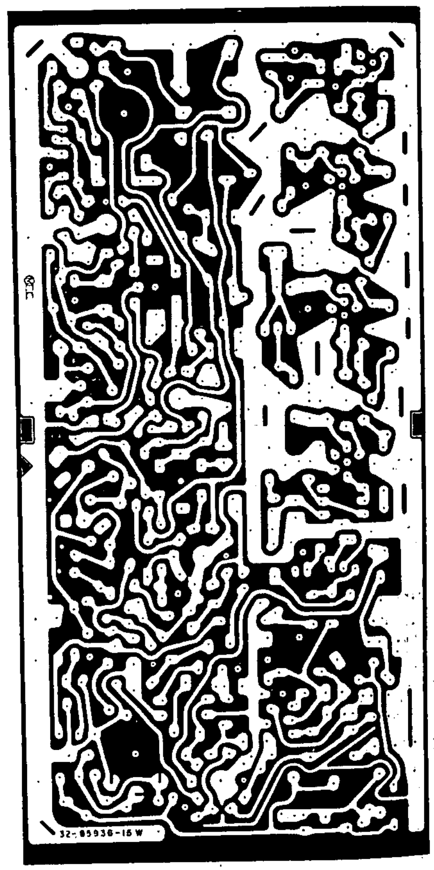
Fig. 31-2. Structural features of copper clad laminate: (A) copper one side, (B) copper two sides.

significant buckling. In its original form, the entire surface of one or both sides of the base is covered with a thin copper sheet that is bonded to the base at all points. This combination of materials is referred to as *copper clad laminate* (Fig. 31-2).

During the manufacture of the commonly used etched printed circuit board, a *resist* solution is used to transfer the appropriate circuit pattern to the surface of the copper clad. A resist is a material that resists the chemical action of an etching solution. The copper clad is then placed in an etching solution containing a chemical such as ferric chloride or chromic acid. After a short period of time, the copper that is not coated with resist is etched or "eaten" away, leaving only those copper surfaces on the board that were coated with resist. Following this, the resist is washed away with a suitable solvent, exposing the copper strip circuit conductor pattern. After mounting holes are drilled or punched in the board, it is ready for component mounting (Fig. 31-3).

For some circuit applications, it is expedient to plate the copper on a printed circuit board with another metal. This is done by an electroplating process either before or after etching. Plating prevents the corrosion of conductor surfaces, thereby making it easier to make solder connections to the conductors.

Fig. 31-3. Etched and punched printed circuit board as it appears before mounting components. (*Eastman Kodak Company*)



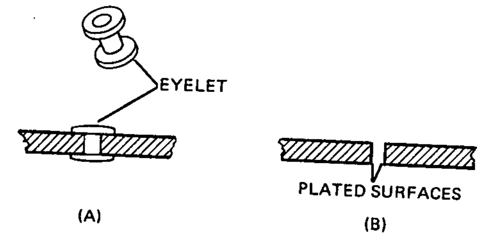


Fig. 31-4. Providing for a conducting path between the sides of a printed circuit board with copper on both sides: (A) using an eyelet, (B) using plated surfaces.

When conductors appear on both sides of a printed circuit board, it is necessary to provide a means of contact between the surfaces at several points of the circuit. This is usually done by means of wire leads or eyelets, or by plating the inner surfaces of holes that are drilled through the board at the points of contact (Fig. 31-4).

Printed circuit wiring has become very popular because it is well adapted to automated mass production manufacturing techniques and to the construction of uniform, small size, lightweight units. Further improvements in the materials used and the ever increasing trend toward more compact (miniaturized) circuit assemblies will no doubt result in an increase in the applications of this process.

THE PRINTED CIRCUIT DIAGRAM

The first step in the manufacture of a printed circuit assembly is to prepare an accurate layout diagram, which is a pattern of the conductors that are to appear upon the baseboard. When it is drawn in ink, such a diagram is usually prepared on a Mylar film or on good quality paper that will not shrink or expand because of humidity or excessive room temperatures. It is very important to use a film or a paper having these properties, since the size of the pattern should remain unchanged over a period of time. The layout pattern is drawn to actual size or to a scale that can be easily reduced, or enlarged photographically to actual size.

Printed circuit diagrams are also often made with tape. In this process, precut strips and pieces of a special tape are assembled into a pattern that conforms with the engineering sketch of a particular circuit (Fig. 31-5).

Drafting Practices. When preparing a printed circuit diagram, the diameter of the holes indicated upon the diagram is usually not less than three-quarters of the thickness of the copper clad to be used. The lines corresponding to the conductors that will appear on the copper clad should not be less than $\frac{1}{32}$ inch wide and should be separated by a distance of at least $\frac{1}{32}$ inch.

Current Carrying Capacity. The maximum current carrying capacity of the printed circuit conductors will of course depend upon the width and thickness of the copper that is bonded to the baseboard. The maximum current carrying capacities of two common thicknesses of copper are given in Table 31-1.

Table 31-1. Typical overload currents for circuits printed upon copper. (*Harris Manufacturing Company*)

Line Width	.00135" Copper Amps	.0027" Copper Amps
$\frac{1}{4}$"	23	35
$\frac{1}{8}$"	15	20
$\frac{1}{16}$"	10	15
$\frac{1}{32}$"	5	8

The Component Overlay. Many types of printed circuit boards have the locations of the components that are to be mounted upon them indicated by symbols or other appropriate markings. This is accomplished by means of a component identification overlay prepared for

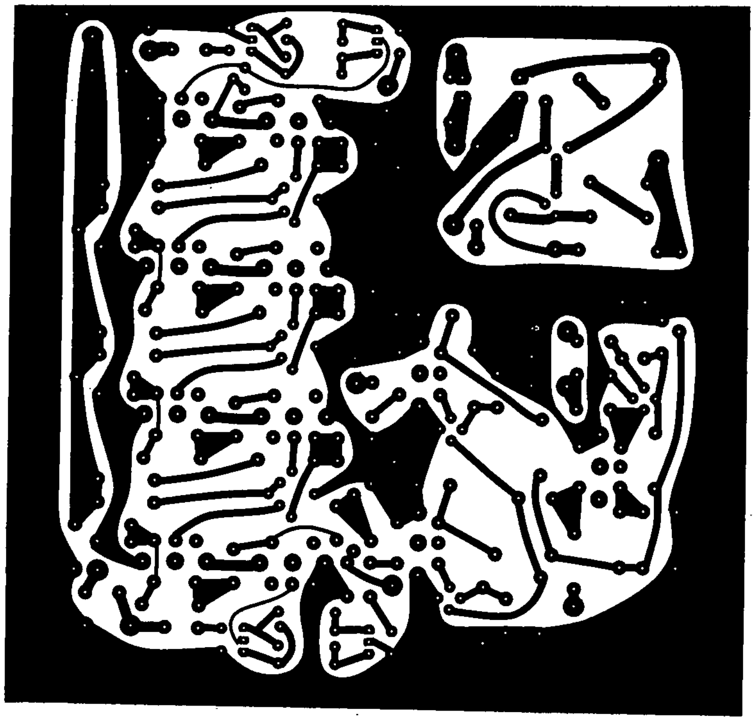


Fig. 31-5. R-F amplifier printed circuit layout diagram made with tape. (*Gates Radio Company, Division of Harris-Intertype Corp.*)

use with a given printed circuit layout diagram (Fig. 31-6). After the printed circuit board has been completed, the identification information indicated upon the overlay is stencilled onto it. This identification of components speeds up the assembly of circuit units and helps in servicing the equipment.

TRANSFER OF THE PATTERN

After the printed circuit diagram has been prepared, it must be transferred to the copper surface of the circuit board. Two commonly used production methods of doing this are the *photoengraving* process and the *silkscreen* process.

Photoengraving Process. The photoengraving process produces a very accurate transfer and is used whenever precision and extremely good detail are required. In this process, the diagram pattern is first photographed and a suitable negative is prepared. The negative is then placed in contact with a copper clad board that has been coated with a light-sensitive substance. The circuit board is next exposed through the negative to a high-energy light source such as a carbon-arc lamp or a mercury-vapor lamp

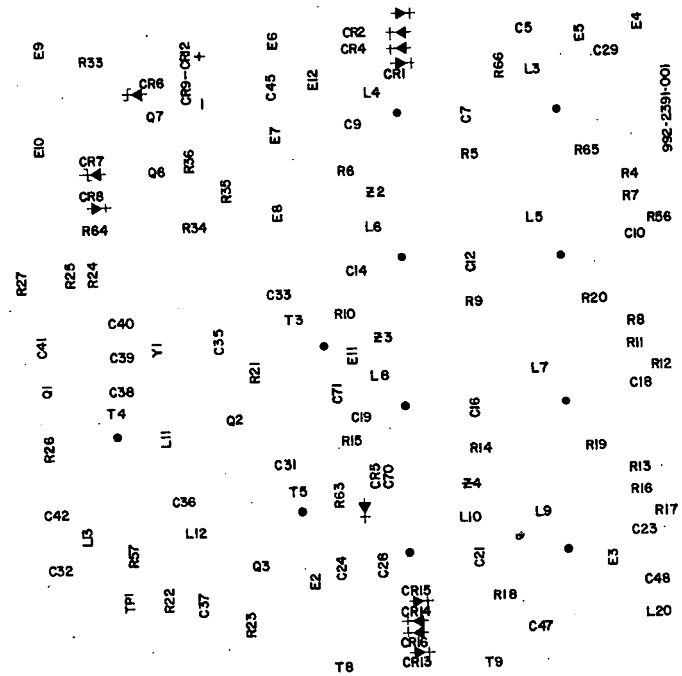


Fig. 31-6. Component identification overlay used in conjunction with the printed circuit layout diagram shown in Fig. 31-5. (*Gates Radio Company, Division of Harris-Intertype Corp.*)

(Fig. 31-7). The light passes through the transparent areas of the negative (the conductor layout) and strikes areas of the copper that are identical in shape to the inked or the taped portions of the layout pattern.

Following exposure, the copper clad is developed. As a result of the development process, an acid-resistant coating, the resist pattern, which is identical to the conductor layout of the printed circuit diagram, is formed on the surface of the copper. Therefore, the copper conductor pattern remaining upon the board after the etching process has been completed is also identical to the conductor layout of the diagram.

Silkscreen Process. The silkscreen process of transferring printed circuit patterns, though less accurate than the photoengraving process, is more adaptable to large quantity, mass-production methods. In the basic silkscreen process, a stencil-type cutout of the circuit conductor pattern is adhered to a silkscreen. The circuit board is then placed under the stencil and a resist compound is applied to the copper through the screen. Because of the stencil, the resist coats only those surface areas of the copper that correspond to the layout pattern. The uncoated copper is then etched from the copper clad.

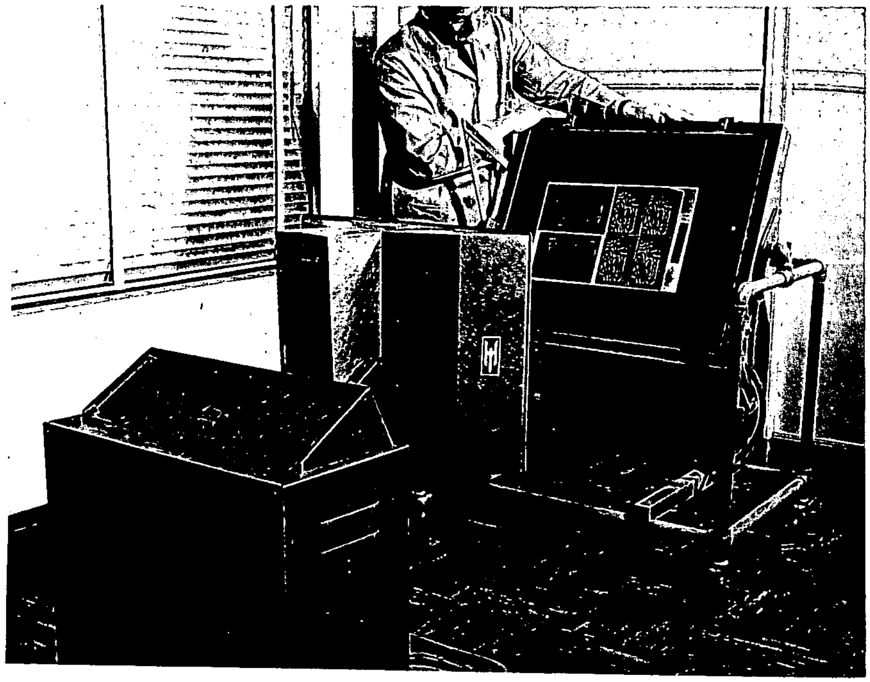
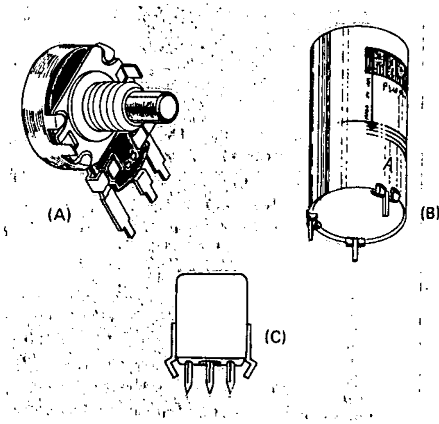


Fig. 31-7. Photoengraving process of printed circuit board fabrication. (*Eastman Kodak Company*)

Fig. 31-8. Printed circuit components: (A) potentiometer, (B) electrolytic capacitor, (C) i-f transformer.



COMPONENT MOUNTING AND SOLDERING

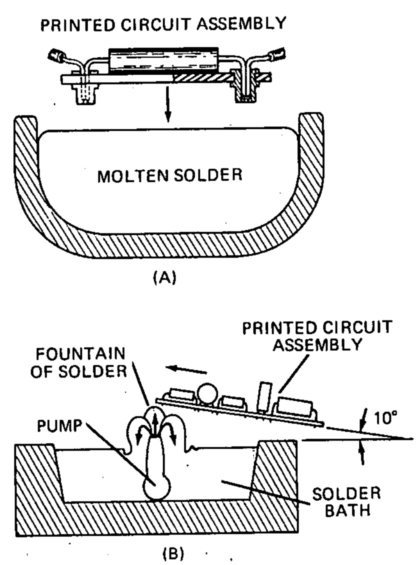
After the printed circuit board is etched and cleaned, unplated conductors are usually coated with a solder-resistant varnish at those points that are not to be soldered. Following this, the components are mounted on the board. The components used with printed circuit wiring are equipped with pin-like terminals to aid in mounting and to contact conductors through holes that are drilled or punched through the board (Fig. 31-8). When the components have been mounted, the assembly is ready for soldering

semblies, the soldering is done by the automated process of either *dip soldering* or *wave soldering*.

Dip Soldering. When dip soldering is used, the surface or surfaces of the wiring side of the printed circuit board are first dipped into a liquid flux. The board is then dipped into molten solder to a depth that is sufficient to allow solder to flow freely into all connection points (Fig. 31-9A).

Wave Soldering. In wave soldering (sometimes referred to as *fountain soldering*), the molten solder is pumped up to the level of the printed circuit board in the form of waves (Fig. 31-9B). This method of soldering, as compared with dip soldering, permits more favorable angles of solder insertion, provides better control for the duration of solder contact, and reduces the amount of heat applied to other parts of the assembly.

Fig. 31-9. Soldering processes used in the manufacture of printed circuit assemblies: (A) dip soldering, (B) wave soldering.



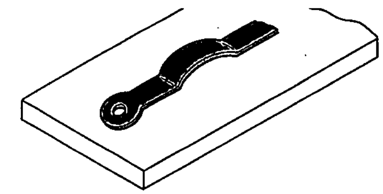


Fig. 31-10. Delaminated printed circuit conductor.

WORKING WITH PRINTED CIRCUITS

The repair and servicing of printed circuit assemblies is not very different from the procedures followed when working with hand-wired circuits. However, because of the nature of the printed circuit and the ease with which it can be seriously damaged by dropping, twisting, overheating, or the excessive application of heat, a more delicate approach is required.

The Soldering Tool. The soldering iron or gun used on a printed circuit board should not have a wattage rating of more than 60 watts. A heavier duty tool, particularly when used by an inexperienced person, produces an excessive amount of heat that may seriously damage the board by burning out conductor strips or by causing the conductors to become separated (delaminated) from the base of the board (Fig. 31-10).

The Solder. Rosin-core solder should be used when working with printed circuits, and the solder should have a comparatively low melting point so that it can be applied with the minimum amount of heat. This condition is satisfied by using a 60-40 (60 percent tin, 40 percent lead) solder with a diameter of approximately $\frac{1}{16}$ inch.

Soldering and Resoldering. When soldering a connecting wire or a component lead to a printed circuit board, it is extremely important not to use any more solder than is absolutely necessary. The use of excessive solder can easily result in shorting out adjacent conductors

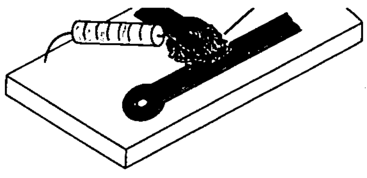


Fig. 31-11. Shorted printed circuit conductors; short caused by use of excess solder.

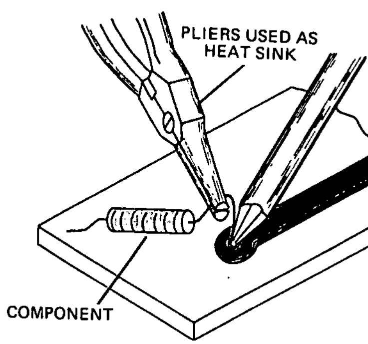
(Fig. 31-11). Attempts to clear up such a short often result in delamination and/or spreading of the solder so that it contacts a greater length of adjacent conductors.

When soldering a semiconductor component to a printed circuit board, its leads should always be left long enough to allow the use of a heat sink (Fig. 31-12). A heat sink should, of course, also be used when unsoldering the lead of a semiconductor component from the board.

Because of the twisting of a printed circuit board or a defective solder connection, it is sometimes necessary to resolder a connection at some point on the board. If at all possible, this should be done by simply reheating the connection without applying more solder.

Repairing Broken Conductors. A simple, "clean" break of a conductor on a printed circuit board

Fig. 31-12. Pliers used as heat sink while soldering a printed circuit component.



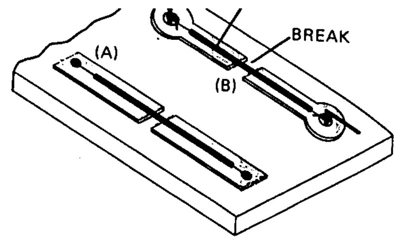


Fig. 31-13. Methods of repairing broken printed circuit conductors.

is most conveniently repaired by carefully flowing (applying) solder across the break. When the break is too large to be handled in this manner, the conductor can usually be repaired by soldering a length of bare, tinned wire across the break (Fig. 31-13A). When the terminals at each end of a broken conductor are readily accessible, it is often desirable to connect hookup wire directly to the terminals themselves (Fig. 31-13B).

Replacing Defective Components. In the majority of cases, the most convenient and safest way of replacing a defective component on a printed circuit board is to avoid disturbing the existing solder connections. When replacing lead-type components such as resistors and capacitors, this is done by first cutting the leads on the component side of the board with diagonal cutting pliers (Fig. 31-14A). The leads of the replacement component are then soldered to the projecting "stubs" of the leads that have been cut (Fig. 31-14B).

When replacing multiterminal components such as transformers, potentiometers, and can-type capacitors, it is best to first cut the terminals. The terminal "stubs" can then be removed from the board one by one. In most cases, the safest way to do this is to grip the terminal with a long-nose plier while heating it at the point where it connects to the board (Fig. 31-14C). After the solder is melted, the terminal is pulled through the mounting hole.

If the terminals on a multiterminal component cannot be cut, it will be necessary to unsolder all connections before the component

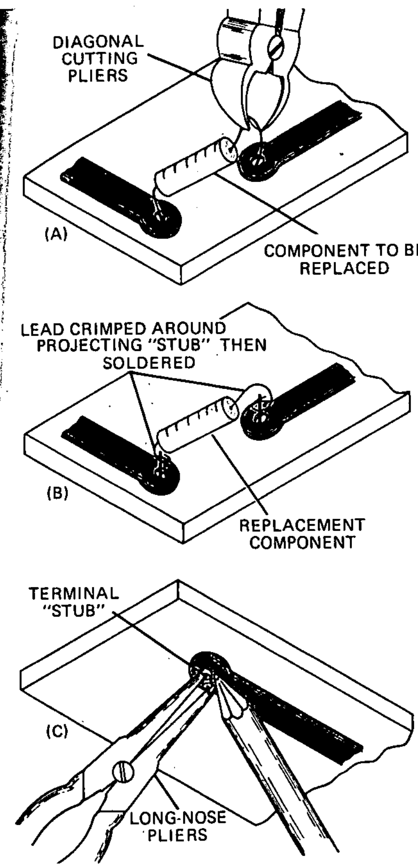


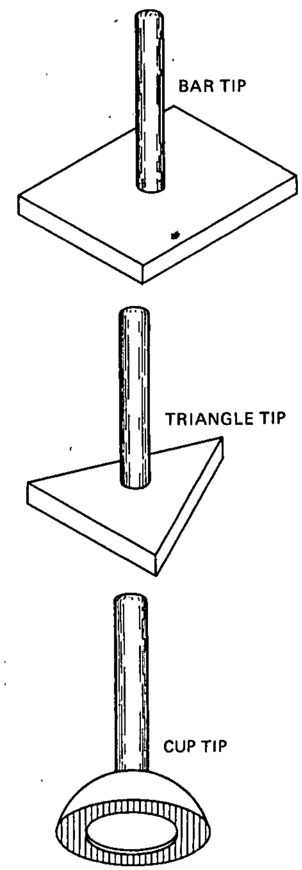
Fig. 31-14. Replacing defective printed circuit components.

can be removed. When doing this, it is best to de-solder all of the connections at the same time. Since it is usually difficult or impossible to perform this operation with an ordinary soldering tip, special de-soldering tips designed for that purpose are used (Fig. 31-15). If a de-soldering tip for a particular job is not available, one can often be made by cutting and bending a relatively heavy copper sheet into the desired

comparatively large surface area, the use of a de-soldering tip may require a heavier duty soldering iron than would be advisable if the tip is not attached to the iron.

In situations where it is impossible to unsolder all of the terminals of a multiterminal component simultaneously, the unsoldering must, of course, be done on a one-by-one basis. The principal problem encountered while doing this is to remove all the melted solder from the

Fig. 31-15. Common types of de-soldering tips.



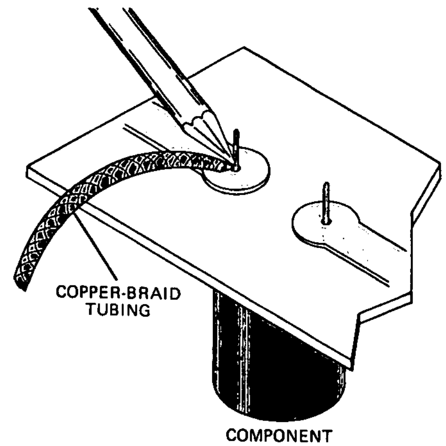


Fig. 31-16. Using copper-braid tubing to remove excess melted solder from a solder connection.

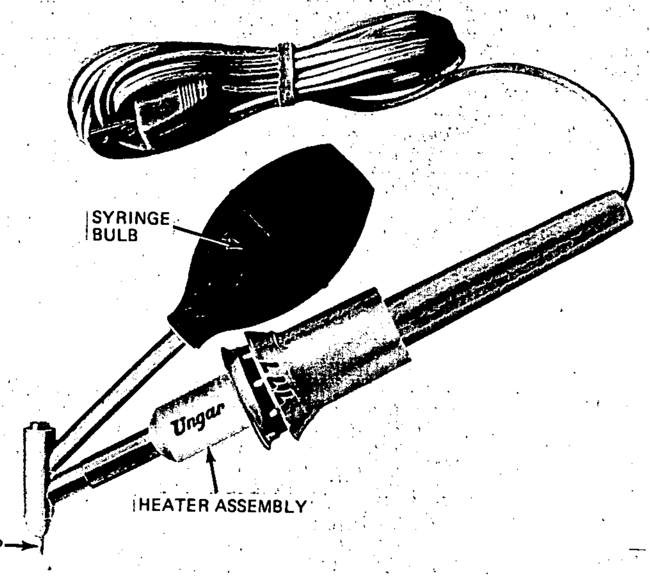
removed, the terminal will become resoldered when the soldering iron or gun is applied to another terminal.

A simple, though not always effective, method of removing melted solder from a given terminal connection point is to use the tip of the soldering iron or gun. If the tip is well tinned, it will attract to itself a small quantity of melted solder, which can then be wiped from the tip. By repeating the process, usually all of the solder can be removed from a connection point.

A more effective way to remove melted solder involves using a length of copper-braid shielding "tubing" as shown in Fig. 31-16. The braid, being in contact with the terminal point being unsoldered, becomes heated and attracts solder, which then flows upward into the braid.

The best job of removing melted solder from a printed circuit board terminal is done by using a syringe-type de-soldering bulb or a similar suction device (Fig. 31-17). When the solder at a given terminal has been melted, the finger

Fig. 31-17. Syringe-type de-soldering tool. (*Ungar Electric Tools*)



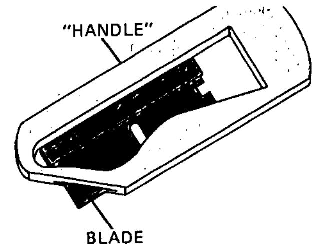


Fig. 31-18. Printed circuit conductor cutting tool.

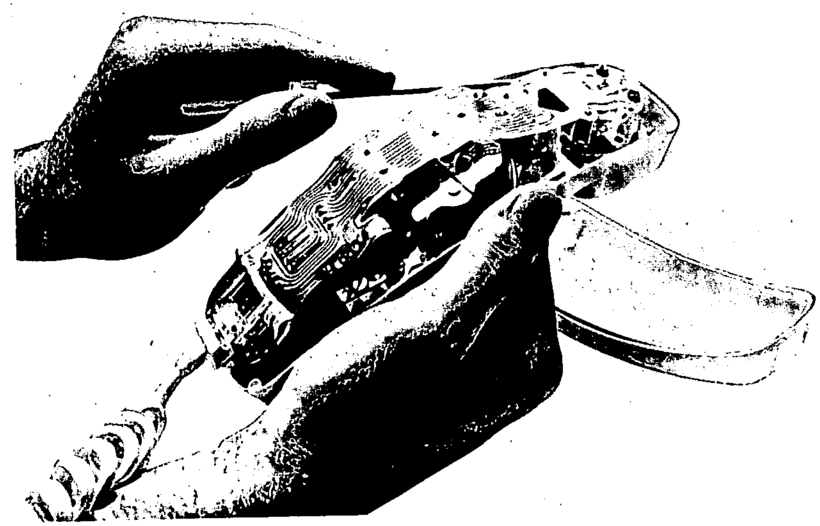
pressure on the bulb of this tool is released, thereby causing solder to be drawn into its hollow tip.

Isolating Components. In making resistance or continuity tests of components on a printed circuit board, it is often desirable to isolate a soldering its lead (or leads) from the associated terminals. This is most conveniently done by cutting the printed circuit conductor (or conductors) connected to the component with a cutting tool (Fig. 31-18). After the tests have been completed, the conductor is "repaired" by flowing solder across the break.

FLEXIBLE CIRCUITRY

The use of flexible copper clad laminate allows a printed circuit to be formed into a variety of shapes, thus enabling the design of circuit structures not possible with rigid materials (Fig. 31-19). In addition to its application as circuit structure material, such laminate is also widely used in making flexible flat cable that often provides for a significant reduction in weight and space as compared with more conventional types of cable.

Fig. 31-19. Flexible copper clad polyester laminate used in circuit of telephone handset. (*Schjeldahl, Electrical Products Division*)



OTHER TYPES OF PRINTED CIRCUITS

Although the etched copper clad laminate printed circuit is the most common type in use today, several additional methods are available for "printing" conductors upon base materials. The scope of this book allows only a brief mention of several of these methods.

The Plated Circuit. The plated circuit board is made by first applying a thin film of a silver compound upon a phenolic laminate plastic base. Those areas of the silver-compound surface that are not to be utilized as conductors are then coated with a resist material. The board is then placed in an electroplating solution, which causes copper (the conductor material) to be plated upon those surfaces not covered with resist.

The Painted Circuit. In the painted circuit process, a liquid containing powdered copper or silver is applied to a base material such as steatite by means of a conductor pattern stencil. This combination of materials is then baked at a temperature which causes the metallic particles to be fused to the base.

Die Stamping. The die-stamped circuit board is produced by first stamping or punching out the desired conductor pattern from a sheet of metal foil. The pattern is then usually attached to a base material by using a suitable adhesive.

Chemical Deposit. In this process of conductor formation, a silver, copper, or other metallic compound solution is deposited upon a base material through an appropriate stencil. The resulting metallic film is then sometimes plated to increase its conductivity.

PRINTED MODULES

An *electronic module* is defined as an assembly of basic components that is used in conjunction with other components to form a complete circuit system. In accordance with this definition, the packaged electronic circuit is one type of module. However, in practice, the module is usually considered to be a more complete assembly that often contains transistors and diodes in addition to the related resistors, capacitors, and inductors. These components are formed by "printing" the essential materials upon individual component or combined component base wafers, which are then interconnected to form the complete unit (Fig. 31-20).

Fig. 31-20. Transistor amplifier module and block schematic diagram of terminal connections. (*Melcor Electronics Corp.*)

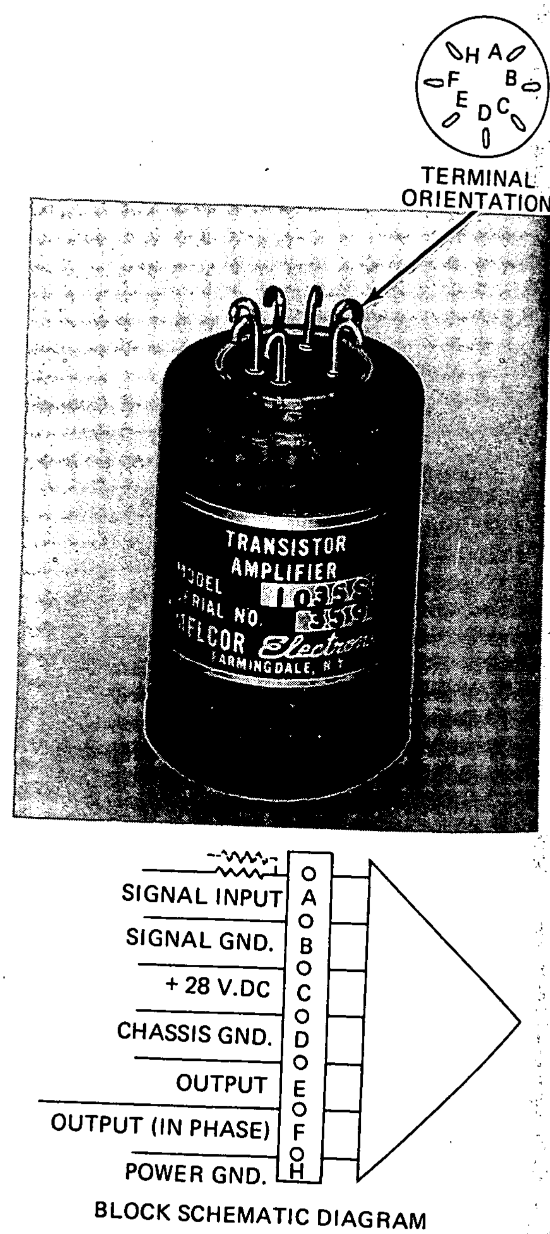


EXHIBIT B

a 250-W iron may be required. It is imperative that the tip of a soldering iron be tinned. A tinned surface has a layer of fresh solder applied to a cleaned portion of the iron. The tinned area can be recognized by its shiny surface in comparison with the brown, oxidized copper of an untinned area. A tinned surface will transfer heat many times faster than an oxidized surface of a hot iron.

Solder that has been heated and cooled several times at a connection oxidizes and crystallizes. It no longer holds properly. If the solder is in this condition, it should be melted from the joint and new solder substituted.

There are other modern means of joining wires to special connector lugs in electric circuits. Crimped connections are produced with a pressure tool that physically crimps the wire and lug together. There is also a special wire-connecting tool that twists the wire around a square terminal lug so tightly that the connection is considered equal to a soldered joint and may be superior if the joint is subject to vibration.

1-17 PRINTED CIRCUITS

All methods of producing a printed circuit (PC) board result in narrow copper-strip conductors laid down on thin insulating sheets of a phenolic or Fiberglas board. The components, or parts, to be connected are then soldered to the copper strips.

One method of producing a PC board is to start with a small oblong of $\frac{1}{16}$-in.-thick phenolic insulator sheet with a thin copper plating on one side. The desired circuit wires are drawn on the copper with a special *resist* ink. The board is then immersed in a ferric chloride solution until all the copper except that under the resist ink has been etched (eaten away) chemically. After this the board is washed and the resist ink removed, leaving a series of copper conductor lines on the phenolic sheet. These may then be silver-plated or plated with solder. Holes are drilled through the copper strips and base. Connector wires from the required electronic components are then slipped into the holes and soldered to the copper lines.

Each connection can be soldered separately, which is rather slow. If all the components are fed through the holes from the insulated side of the board, all the connections can be *wave-soldered* at once by moving the copper-connection surface of the PC board over a wave of molten solder. The wave is developed over a cylinder rotating just under the surface of a small tub full of molten solder.

One of the difficulties with printed circuits stems from the difference of expansion of the copper and the phenolic boards. Temperature changes can loosen the copper strips from the phenolic backing, or hairline cracks can open up across the strips, breaking the electrical connections. This may occur during the original soldering, if heated during operation, or during unsoldering to remove and replace faulty components. Bending the boards may develop similar troubles. While resistance meters (ohmmeters) can be used to determine if a circuit has broken open, visual examination with a powerful magnifying glass will often be simpler.

When components are removed from a printed circuit, a very small soldering iron must be used to prevent overheating of the PC board. A pulling tension is applied to one end of the lead of the component being removed while a 25- to 35-W soldering iron tip is applied to the soldered contact. As the solder melts, the component wire pulls free of the PC board hole. If the hole remains plugged with solder, it may be heated and a stainless-steel wire, sharp lead pencil, or other thin device to which solder will not adhere can be run into the hole to clear it. A far better method is to use a special small soldering iron called a *desoldering tool* with a hollow tip to which is coupled a rubber bulb. The bulb is squeezed, forcing air out the hollow tip. The hot tip is then placed around the wire to be desoldered until the solder melts. When the bulb is released, solder is sucked up into the hollow iron, freeing the component lead of solder and allowing the lead to be extracted. Squeezing the bulb again forces the molten solder out of the hollow tip, readying the iron for use again.

One commercial process of producing printed circuit boards is *photoetching*. The original circuit is drawn on paper and photographed. The negative is then used to transfer, by photographic methods, an etch-resistant image of the wiring onto a photographically surfaced copper-clad

phenolic board. The rest of the method is as described above. Other methods used are stenciling, chemical deposition, and vacuum distillation.

Test your understanding; answer these checkup questions.

1. How many hertz in a kilohertz? ________ In a megahertz? ________
2. How many milliamperes in 2.45 A? ________ In 0.0358 A? ________
3. How many microamperes in 29 mA? ________ In 7.2 A? ________
4. How many amperes in 450 mA? ________ In 75 μA? ________
5. How many milliwatts in 850 μW? ________ In 5.6 W? ________
6. From Table 1-3, how many ohms does 100 ft of No. 10 copper wire have? ________ No. 20? ________
7. Using 1,000 cmil/A, what size wire is required for a 4-A flow? ________
8. Using 400 cmil/A, what size wire is required for a 1-A flow? ________ A 4-A flow? ________
9. At what temperature does a good solder melt? ________
10. What is the reason for soldering a wire splice? ________
11. What is the duty of solder in PC board connections? ________
12. What is the duty of solder fluxes? ________
13. With what may circuit connections be drawn on PC boards before they are chemically treated? ________
14. What wattage iron would be used on an antenna wire outdoors? ________ On a PC board? ________ On indoor electrical wiring? ________
15. What is the coating of solder on the tip of a hot soldering iron called? ________
16. What method of soldering can make a hundred soldered connections at once on PC boards? ________
17. What is the best device to use to remove solder from PC board connections? ________

COMMERCIAL LICENSE QUESTIONS

Amateur license questions will be found in the Addendum.

An ability to answer questions similar to the following ones is required for FCC Elements 3, 4, and 6. A question followed by a bracketed number is required for that element alone. Sections in which questions are answered are shown in parentheses.

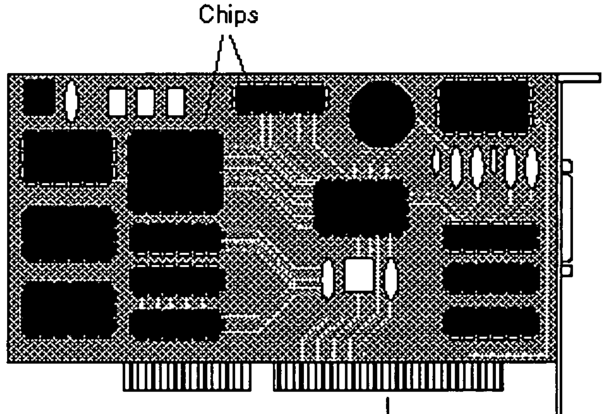
1. What is an *electron?* (1-2)
2. Explain the relationship between the physical structure of the atom and electric-current flow. (1-3, 1-7)
3. In what manner does the resistance of a copper conductor vary with variations in temperature? (1-3, 1-12)
4. Define the term *coulomb.* (1-7)
5. By what other expression may *electric-current flow* be described? (1-7)
6. By what other expression may a *difference of potential,* or *electromotive force,* be described? (1-8)
7. What is an *ion?* (1-10)
8. What is the unit of resistance? (1-12)
9. With respect to electrons, what is the difference between conductors and nonconductors? (1-12)
10. If the diameter of a conductor of given length is doubled, how will the resistance be affected? (1-12)
11. Explain the factors which affect the resistance of a conductor. (1-12)
12. What is the relationship between wire size and resistance of the wire? (1-12)
13. Name four conducting materials in their order of conductivity. (1-12)
14. What would be the value and tolerance of a resistor if color-coded, from left to right, green, brown, yellow, silver? (1-13)
15. What would be the value and tolerance of a resistor if color-coded, from left to right, red, black, orange, gold? (1-13)
16. Explain the meaning of *kilo, micro, mega, pico, micromicro, nano.* (1-14)
17. Make the following transformations: (*a*) kilohertz to hertz, (*b*) kilovolts to volts, (*c*) milliamperes to amperes. (1-14)
18. List at least two essentials for a good soldered connection. (1-16)
19. List three precautions which should be taken in soldering electrical connections to assure a permanent junction. (1-16)
20. Why is rosin used as soldering flux in radio construction work? (1-16)
21. Discuss etched-wiring printed circuits with respect to determination of wiring breaks, excessive heating, and removal or installation of components. (1-17)

NOTE: FCC license test questions are of the multiple-choice type, taking a form somewhat as follows:

EXHIBIT C

printed circuit board

Last modified: Wednesday, November 12, 2003



Sometimes abbreviated *PCB,* a thin plate on which chips and other electronic components are placed. Computers consist of one or more boards, often called cards or *adapters*. Circuit boards fall into the following categories:

- **motherboard** : The principal board that has connectors for attaching devices to the bus. Typically, the mother board contains the CPU, memory, and basic controllers for the system. On PCs, the motherboard is often called the *system board* or *mainboard.*
- **expansion board** : Any board that plugs into one of the computer's expansion slots. Expansion boards include controller boards, LAN cards, and video adapters.
- **Daughtercard** : Any board that attaches directly to another board.
- **controller board:** A special type of expansion board that contains a controller for a peripheral device. When you attach new devices, such as a disk drive or graphics monitor, to a computer, you often need to add a controller board.
- **Network Interface Card (NIC)** : An expansion board that enables a PC to be connected to a local-area network (LAN).
- **video adapter:** An expansion board that contains a controller for a graphics

printed circuit board —a flat plate or base of insulating material containing a pattern of conducting material. It becomes an electrical circuit when components are attached and soldered to it.

> The conducting material is commonly copper which has been coated with solder or plated with tin or tin-lead alloy. The usual insulating material is epoxy laminate. But there are many other kinds of materials used in more exotic technologies.
>
> Single-sided boards, the most common style in mass-produced consumer electronic products, have all conductors on one side of the board. With two-sided boards, the conductors, or copper traces, can travel from one side of the board to the other through plated-thru holes called vias , or feed-throughs. In multilayer boards, the vias can connect to internal layers as well as either side

This Page is Inserted by IFW Indexing and Scanning Operations and is not part of the Official Record

BEST AVAILABLE IMAGES

Defective images within this document are accurate representations of the original documents submitted by the applicant.

Defects in the images include but are not limited to the items checked:

- ☐ BLACK BORDERS
- ☐ IMAGE CUT OFF AT TOP, BOTTOM OR SIDES
- ☐ FADED TEXT OR DRAWING
- ☐ BLURRED OR ILLEGIBLE TEXT OR DRAWING
- ☐ SKEWED/SLANTED IMAGES
- ☐ COLOR OR BLACK AND WHITE PHOTOGRAPHS
- ☐ GRAY SCALE DOCUMENTS
- ☑ LINES OR MARKS ON ORIGINAL DOCUMENT
- ☐ REFERENCE(S) OR EXHIBIT(S) SUBMITTED ARE POOR QUALITY
- ☐ OTHER: ______________________

IMAGES ARE BEST AVAILABLE COPY.
As rescanning these documents will not correct the image problems checked, please do not report these problems to the IFW Image Problem Mailbox.